3D BEAM MANUAL

Sangwook Sihn, Stephen W. Tsai
Department of Aeronautics & Astronautics
Stanford University, CA 94305-4035
Phone : (650) 725-3305
Fax : (650) 725-3377
email : sangwook@structure.stanford.edu, stsai@structure.stanford.edu

1 Introduction

Many sub-parts of complicated structures can be simplified as beams or frames, if the length is relatively larger than the size of their cross-section. Examples of these substructures are the airfoil of airplanes; hull mast and keel of ships; rotor, golf shaft, etc. Consequently, the analysis of beam elements is the key step in analyzing the whole structure.

Nowadays, composite materials are used in many structures because of their lightness and superiority in their stiffness and strength. Analysis of the laminated composite structures needs ply-by-ply inputs, such as thickness and orientation as well as the material properties. Not only are the ply-by-ply inputs cumbersome, but it is not easy to change the parameters for the design purpose, especially with very complicated cross-sections.

Analysis of the beams as one-dimensional structure is advantageous in terms of its speed and simplicity. However, many anaylsis tools including the commercial finite element packages, such as Ansys, Abacus and Nastran, have 1-dimensional beam elements only for isotropic materials, but not for the composites. We need to use either 2-dimensional plate or 3-dimensional shell elements for the laminated composite structures.

3D_BEAM is a finite element program for analyzing the composite beam and frame structures with arbitrary cross-sections by the one-dimensional approach. It is a spreadsheet based program, running on Microsoft Excel, to make user-friendly input and analysis possible. The program requires

1) PC with Microsoft Windows or Macintosh

2) Microsoft Excel 95 or higher

2 General description

2.1 Coordinate systems

Users note that **3D_Beam** uses three coordinate systems: global coordinate system (denoted by X, Y, Z), reference coordinate system (denoted by X_1-axis, X_2-axis, X_3-axis), and local coordinate system (denoted by x_1-axis, x_2-axis, x_3-axis).

The global and reference coordinate systems are shown in Figure 1. A beam element is defined by two nodes (I and J). The reference X_1-axis is oriented from node I toward node J. The default orientation ($\theta = 0°$) of the reference X_2-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the reference X_2-axis is oriented parallel to the global Y-axis.

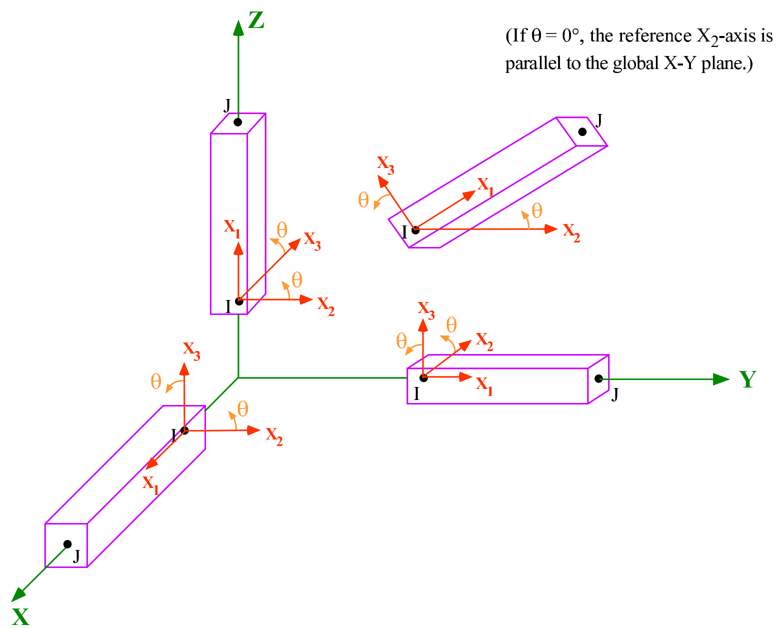


Figure 1: Global and reference coordinate systems.

For user control of the element orientation about the reference X_1-axis, use the θ angle. The local coordinate system (x_1-x_2-x_3) is then defined by rotating the reference coordiate system with the angle, θ, with respect to the X_1-axis. This local coordinate system may have multiple definitions along a reference X_1-axis. Figure 2 shows reference and local coordinate systems.

It is convenient to prepare beams with complicated cross sections (e.g., airfoil shape) in this local coordinate system. If a simple cross-section beam (e.g., circular, elliptic, boxed) is to be analyzed, let the local coordinate system be simply the reference coordinate system, and the rotation angle, θ, be zero.

2.2 Element, node and element group

Beams are divided into several pieces along the reference X_1-axis by a suitable length for the finite element analysis. Such a piece is called an *element*. the element is called a *node*.

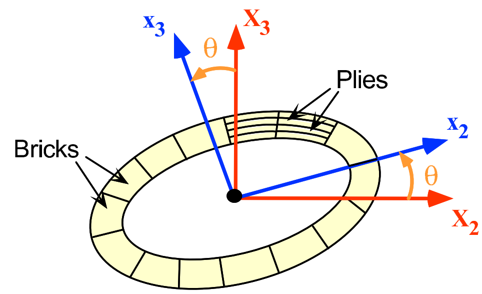


Figure 2: Reference and local coordinate systems.

Prepare the **global** coordinates of all nodes for the finite element analysis.

The collection of elements that have the same cross-sectional information (layup, thickness, material, etc) and the same local coordinate system is called an *element group*. Figure 3 shows elements, nodes and element groups.

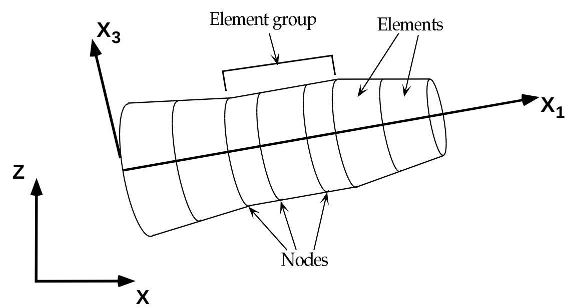


Figure 3: Elements, nodes and element groups.

2.3 Directions of six degree of freedoms and forces

It is important to know the direction of displacements, rotations, forces and moments to understand the result of the finite element analysis correctly. The direction of six degrees of freedoms are shown in Figure 4. Note that these are defined in the **global** coordinate system.

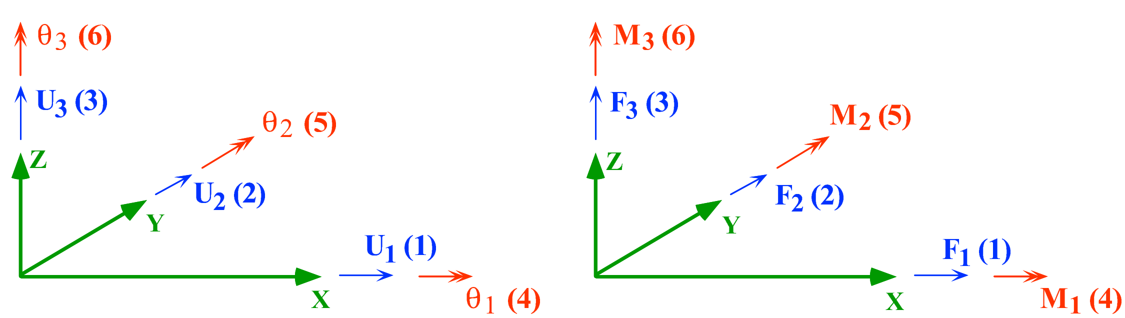


Figure 4: Direction of six degree of freedoms and forces.

2.4 Brick, brick node and ply

The cross-section of an element is divided into several pieces, and such a piece is called a *brick*. Each of the end points of the brick are called a *brick node*. Prepare the coordinates of all brick nodes in the **local** coordinate system. If the beam is made of the composite laminate, you'll see many plies in each brick. Figure 5 shows bricks, brick nodes and plies.

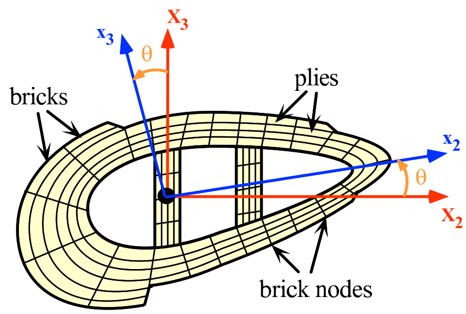


Figure 5: Bricks, brick nodes and plies.

2.5 Ply orientations

The finite element analysis requires information about the cross-sections such as the geometry and the material properties. The geometric information is known by specifying the local coordinates of the brick nodes as explained in Section 2.4.

The information for the material properties is known by specifying the Young's modulus and the Poisson's ratio. In addition, if the composite beam is to be analyzed, you need to know the orientation of the fibers of each ply. This orientation is called *ply orientation*.

Ply orientation (ϕ) is determined as the angle between the local x_1-axis and the direction of the fiber. A positive angle is the angle from the local x_1-axis to the orientation of the fiber in a clockwise direction, looking at the center of the cross-section from a point outside the beam. Figure 6 shows a detailed view of the ply whose fiber orientation is the positive ϕ.

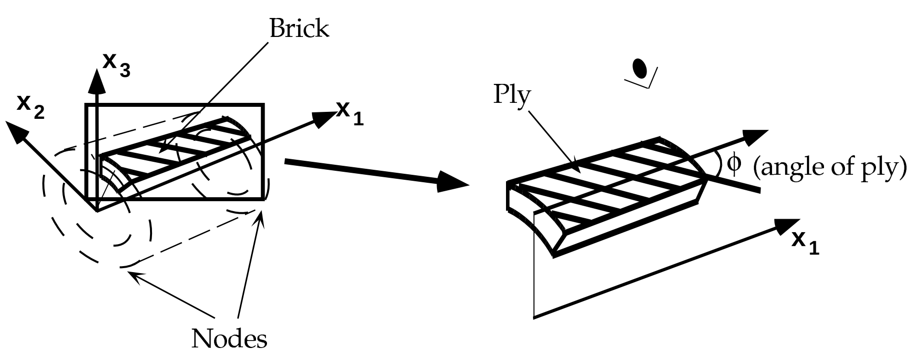


Figure 6: Ply orientation.

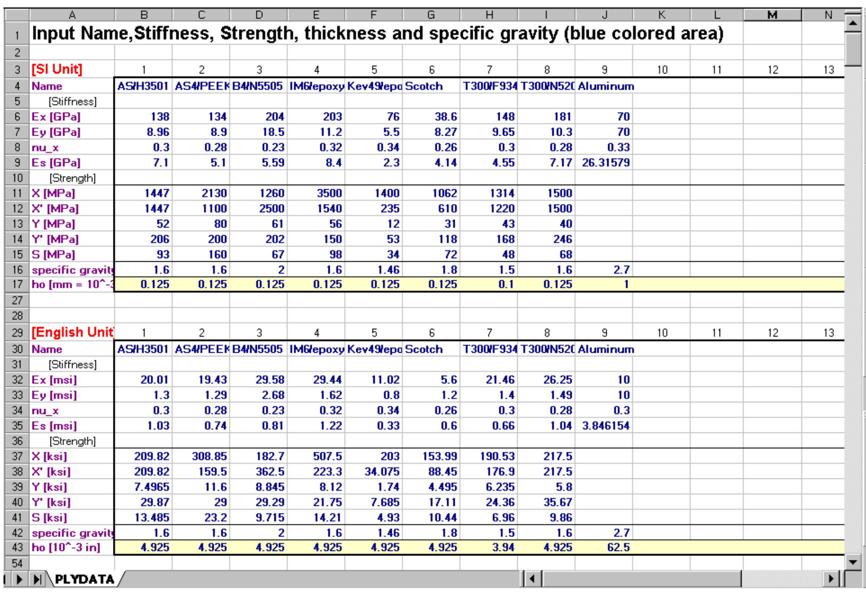
2.6 Material property input

Eight materials are pre-inputted in the file **PLYDATA.XLS**. The corresponding numbers for these materials are listed in Table 1.

Material #	Name	Material #	Name
1	AS/H3501	5	Kev 49/epoxy
2	AS4/PEEK	6	Scotch
3	B4/N5505	7	T300/F934
4	IM6/epoxy	8	T300/N5208

Table 1: Material numbers

To input a new material or change the material property, go to **Unhide** menu under **Window** menu, then select the file **PLYDATA.XLS**. Input new or changed material constants (e.g., engineering constants, thickness of ply, etc.) into the file, as in Figure 7. You can input up to 20 materials.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Input Name,Stiffness, Strength, thickness and specific gravity (blue colored area)													
2														
3	[SI Unit]	1	2	3	4	5	6	7	8	9	10	11	12	13
4	Name	AS/H3501	AS4/PEEK	B4/N5505	IM6/epoxy	Kev49/epo	Scotch	T300/F934	T300/N52(	Aluminum				
5	[Stiffness]													
6	Ex [GPa]	138	134	204	203	76	38.6	148	181	70				
7	Ey [GPa]	8.96	8.9	18.5	11.2	5.5	8.27	9.65	10.3	70				
8	nu_x	0.3	0.28	0.23	0.32	0.34	0.26	0.3	0.28	0.33				
9	Es [GPa]	7.1	5.1	5.59	8.4	2.3	4.14	4.55	7.17	26.31579				
10	[Strength]													
11	X [MPa]	1447	2130	1260	3500	1400	1062	1314	1500					
12	X' [MPa]	1447	1100	2500	1540	235	610	1220	1500					
13	Y [MPa]	52	80	61	56	12	31	43	40					
14	Y' [MPa]	206	200	202	150	53	118	168	246					
15	S [MPa]	93	160	67	98	34	72	48	68					
16	specific gravit	1.6	1.6	2	1.6	1.46	1.8	1.5	1.6	2.7				
17	ho [mm = 10^-3	0.125	0.125	0.125	0.125	0.125	0.125	0.1	0.125	1				
27														
28														
29	[English Unit	1	2	3	4	5	6	7	8	9	10	11	12	13
30	Name	AS/H3501	AS4/PEEK	B4/N5505	IM6/epoxy	Kev49/epo	Scotch	T300/F934	T300/N52(	Aluminum				
31	[Stiffness]													
32	Ex [msi]	20.01	19.43	29.58	29.44	11.02	5.6	21.46	26.25	10				
33	Ey [msi]	1.3	1.29	2.68	1.62	0.8	1.2	1.4	1.49	10				
34	nu_x	0.3	0.28	0.23	0.32	0.34	0.26	0.3	0.28	0.3				
35	Es [msi]	1.03	0.74	0.81	1.22	0.33	0.6	0.66	1.04	3.846154				
36	[Strength]													
37	X [ksi]	209.82	308.85	182.7	507.5	203	153.99	190.53	217.5					
38	X' [ksi]	209.82	159.5	362.5	223.3	34.075	88.45	176.9	217.5					
39	Y [ksi]	7.4965	11.6	8.845	8.12	1.74	4.495	6.235	5.8					
40	Y' [ksi]	29.87	29	29.29	21.75	7.685	17.11	24.36	35.67					
41	S [ksi]	13.485	23.2	9.715	14.21	4.93	10.44	6.96	9.86					
42	specific gravit	1.6	1.6	2	1.6	1.46	1.8	1.5	1.6	2.7				
43	ho [10^-3 in]	4.925	4.925	4.925	4.925	4.925	4.925	3.94	4.925	62.5				
54														

PLYDATA

Figure 7: **PLYDATA.XLS**: Data file for material property input.

3 Flow chart

There are three steps in **3D_BEAM** for the finite element analysis. These are **pre-processor**, **main-solver**, and **post-processor**, as shown in Figure 8.

The **pre-processor** calculates $[A], [B], [D]$ matrices from the data given in the cross-section. These matrices are used in formulating the stiffness matrix of the beam elements.

The **main-solver** calculates the displacements and rotations using the $[A], [B], [D]$ matrices, as well as the coordinates and boundary conditions, which are prepared in files **ELEM.XLS** and **NODE.XLS**, This step also calculates the weight of the structure.

The **post-processor** does two jobs consecutively. The first job is to calculate the mid-surface strains and curvatures and the resultant forces at each node in the **global** coordinate system. The second job is to calculate the strains and the stresses at each ply in the on-axis coordinate system.

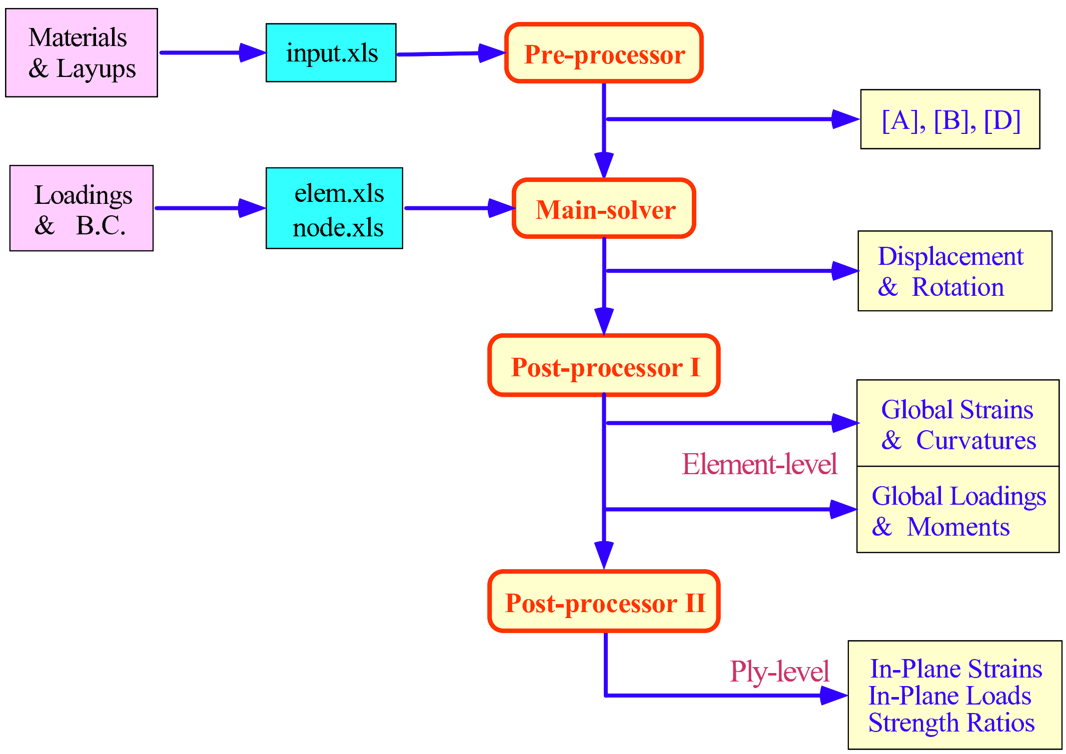


Figure 8: Solution procedure of **3D_BEAM**.

4 How to start?

There are 12 files in the distributed diskette. They are **3D_BEAM.XLW**, **BEAM.XLM**, **BEAMK.XLS**, **ELEM.XLS**, **FACTOR.XLM**, **INPUT.XLS**, **NODE.XLS**, **OUTPUT.XLS**, **PLYDATA.XLS**, **POST.XLS**, **PRE.XLS** and **PRE-POST.XLM**.

Run or open the file **3D_BEAM.XLW**, you will see only 3 files—**INPUT.XLS**, **ELEM.XLS** and **NODE.XLS**. Other files are hidden when they are opened.

4.1 Pre-processor

The file **INPUT.XLS** plays two roles—one for the input for the **pre-processor** and the other for the **control** of this program. To input the information such as the local coordinates and the ply orientations, etc., move to the right of this file so that you can see Figure 9.

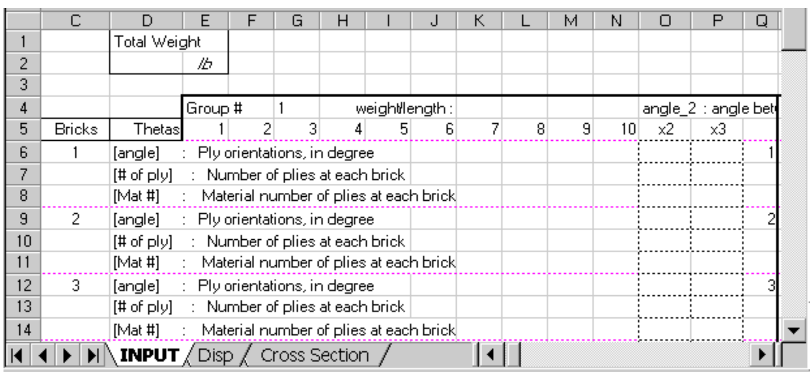


Figure 9: Input file for **pre-processor**: **INPUT.XLS**

Figure 9 shows the input boxes for the information for the cross-section for up to 19 element groups and up to 30 bricks. The required inputs are

a) the local x_2-axis and x_3-axis coordinates of the brick nodes in meters or inches.

b) the angle (θ) between the reference coordinate system and the local coordinate system in a cell **P4**.

c) the orientations and the numbers of the plies in each brick. The orientations must be in degree.

c) the material numbers of the plies in each brick.

After filling in all the information you need, move to the left. Then input the numbers for control, as explained in Figure 10.

To confirm the geometry of the cross-section, click "**Show**" button. To start the **pre-processor**, click "**Start Pre-process**" button. The $[A], [B], [D]$ matrices are calculated and recorded in the hidden file **BEAMK.XLS**.

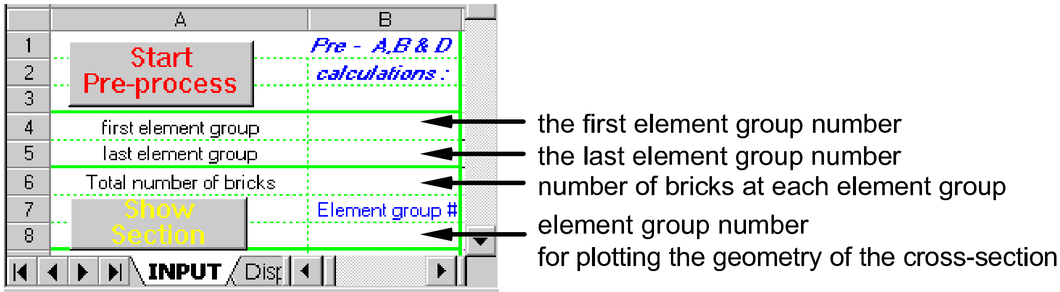


Figure 10: Control part for the **pre-processor**: **INPUT.XLS**

4.2 Main-solver

The two files **NODE.XLS** and **ELEM.XLS** need filling out with the global coordinates, boundary conditions and nodal connectivity. First, input the followings in the file **NODE.XLS**, as explained in Figure 11:

a) nodal coordinates in the **global** coordinates system in meters or inches.

b) boundary conditions (ID) at the nodal points:

  * Set ID be 0 for free nodes.

  * Set ID be 1 for restrained nodes.

  ** Warning : Be careful not to have any rigid body motion.

c) nodal forces and moments in the **global** coordinate system in **Mega** Newtons or **Kilo** pounds and in MN-m or klbf-in, respectively.

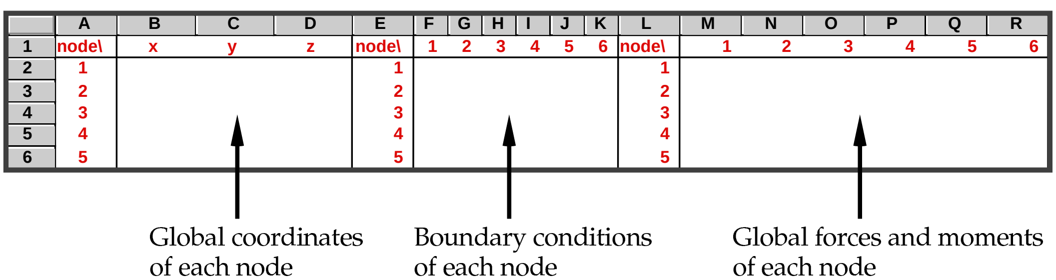


Figure 11: Input file for the **main-solver**: **NODE.XLS**

Then select the file **ELEM.XLS** and input the followings, as explained in Figure 12:

a) nodal connectivities of the elements.

b) element group numbers prepared during the **pre-process**.

Once you have filled in all the information you need, go back to the file **INPUT.XLS**. Then answer the questions and input the number of nodes and the number of elements, as explained in Figure 13.

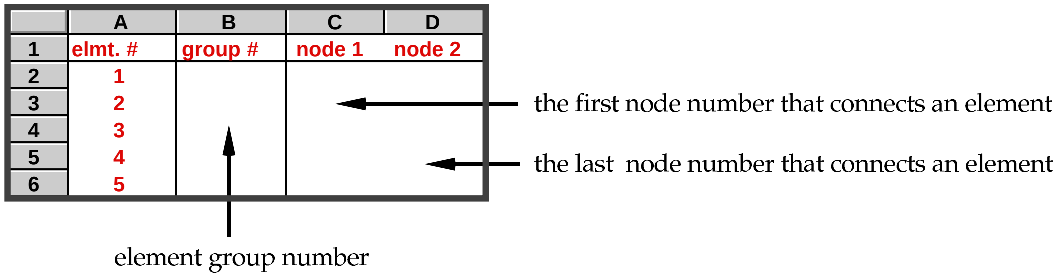


Figure 12: Input file for the **main-solver**: **ELEM.XLS**

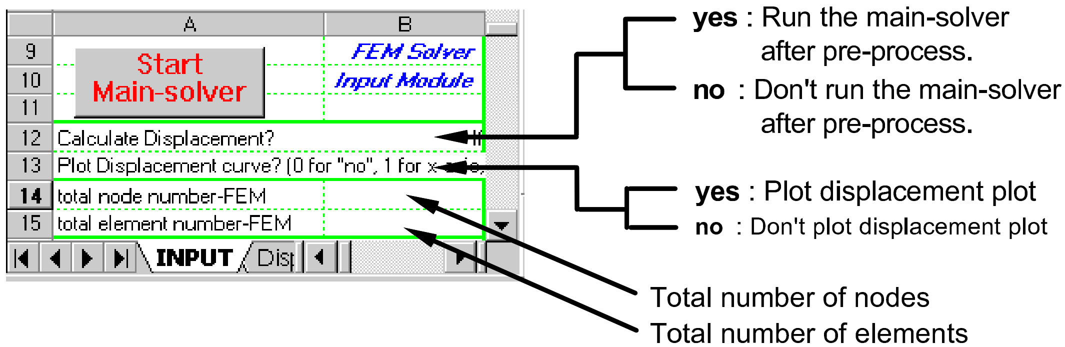


Figure 13: Control part for the **main-solver**: **INPUT.XLS**

Start the **main-solver** by press "**Start Main-solver**" button. It calculates the displacements and rotations in the **global** coordinate system. You can plot them by inputting **"yes"** in the cell **B13** (see Figure 13). These results are saved in a worksheet **Disp** in the file **INPUT.XLS**. You will also see the weight of the structure in the cell **D2** in the worksheet **INPUT** in the same file.

4.3 Post-processor

Once the **main-solver** calculates the displacement, you can do the **post-processor** by answering the questions and inputting the element numbers into the file **INPUT.XLS**. This **post-processor** is divided into two parts.

The first part requires two simple answers as explained in Figure 14.

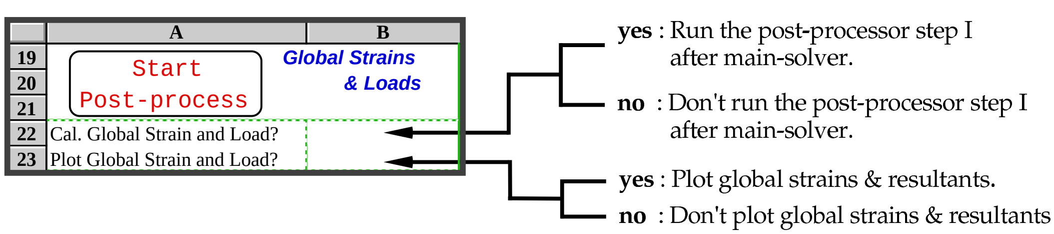


Figure 14: Control part for the **post-processor** step I: **INPUT.XLS**

If you press "**Start Post-process**" button, you will have the mid-plane strains and curvatures in the **global** coordinate system. You'll also have the global in-plane forces and moments. You can plot them by inputting **“yes”** in the cell **B20** (see Figure 14). The results are saved in the hidden file **FACTOR.XLM**.

The second part requires filling out the remaining parts in the file **INPUT.XLS** as explained in Figure 15. Type **“strain”** or **“stress”** in the cell **B26**. This cell specifies the option for the plane strain/plane stress assumption. Choose plane strain if the beam has a closed cross-section (e.g., airfoil, circular, elliptic cross-section beam, etc.). Choose plane stress in other cases.

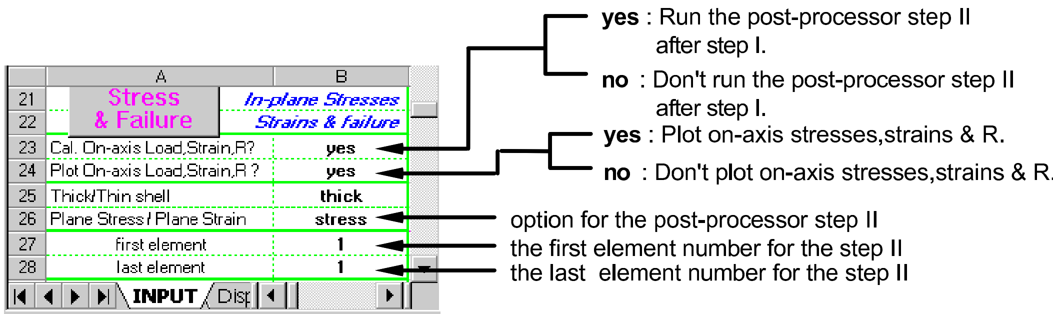


Figure 15: Control part for the **post-processor** step II: **INPUT.XLS**

Click “**Stress & Failure**” button to calculate the in-plane (on-axis) strains, stresses and the failure indices ($k = 1/R$, R : Strength ratio). You can plot them by inputting **“yes”** in the cell **B24** (see Figure 15). The results are saved in the file **OUTPUT.XLS**.

4.4 New run

If you want to run other cases, change the parameters you need and go back to either Section 4.1 or Section 4.2. If you want to change the information of the cross-sections representing the element groups, go back to Section 4.1 and follow the subsequent steps. If you don't want to change the orientations and the layup and the material of the plies, but just want to change the nodal coordinates or the boundary conditions or the global forces and moments, go back to Section 4.2 and follow the subsequent steps.

5 Examples

5.1 Case 1

The first case is a circular cross-sectional cantilever. One end is fixed and the other end is free. The length of this beam is 50 cm, and the diameter of the cross-section is 3 cm. Concentrated force (.01 MN) is applied downward at the free end. Figure 16 shows this beam and its cross-section.

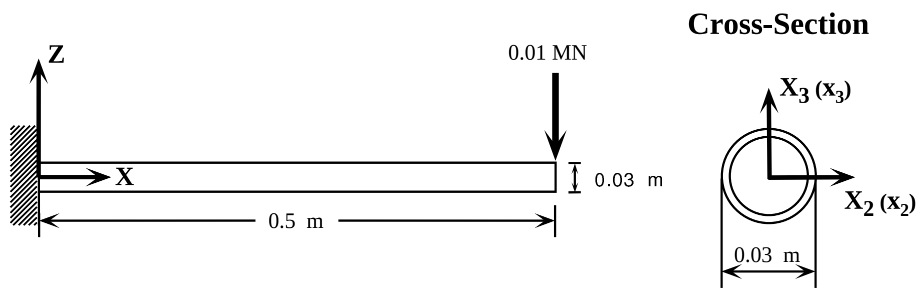


Figure 16: Simply-supported beam with a circular cross-section subjected to concentrated force

Now input numbers on the right side of the file **INPUT.XLS** as shown in Figure 17. To make the circular cross-section, do the following:

* input the radius of the circle in the cell **Q1**.
* input the following formula in the cell **O6** to have the value 0.015 ; "$= \$Q\$1 * COS(18 * (\$Q6 - 1) * PI()/180)$".
* input the following formula in the cell **P6** to have the value 0.000 ; "$= \$Q\$1 * SIN(18 * (\$Q6 - 1) * PI()/180)$".
* input the following formula in the cell **O7** to have the value 0.014 ; "$= \$Q\$1 * COS(18 * \$Q6 * PI()/180)$".
* input the following formula in the cell **P7** to have the value 0.005 ; "$= \$Q\$1 * SIN(18 * \$Q6 * PI()/180)$".
* Copy the cells **O6:P8** to **O9:P65**.

	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q
1		Total Weight												radius	0.015
2		*kg/lbf*													
3															
4			**Group #**	**1**	**weight/length :**								**angle_2**	**0.000**	
5	Bricks	Thetas	1	2	3	4	5	6	7	8	9	10	x2	x3	
6	1	[angle]	**0**	**45**	**-45**	**90**	**90**	**-45**	**45**	**0**	**0**	**0**	**0.015**	**0.000**	**1**
7		[# of ply	**3**	**1**	**1**	**1**	**1**	**1**	**1**	**3**	**0**	**0**	**0.014**	**0.005**	
8		[Mat #]	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**0**	**0**			
9	2	[angle]	**0**	**45**	**-45**	**90**	**90**	**-45**	**45**	**0**	**0**	**0**	**0.014**	**0.005**	**2**
10		[# of ply	**3**	**1**	**1**	**1**	**1**	**1**	**1**	**3**	**0**	**0**	**0.012**	**0.009**	
11		[Mat #]	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**0**	**0**			
12	3	[angle]	**0**	**45**	**-45**	**90**	**90**	**-45**	**45**	**0**	**0**	**0**	**0.012**	**0.009**	**3**
13		[# of ply	**3**	**1**	**1**	**1**	**1**	**1**	**1**	**3**	**0**	**0**	**0.009**	**0.012**	
14		[Mat #]	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**8**	**0**	**0**			

Figure 17: Inputs for the **pre-processor**

Once you finish inputting these coordinates, check out the geometry of the cross-section in the reference coordinate system to see if you inputted them correctly. Go to the control part of the file **INPUT.XLS** and press the "**Show**" button. You'll see the plot of the circle composed of the brick nodes.

Figure 18 shows this plot.

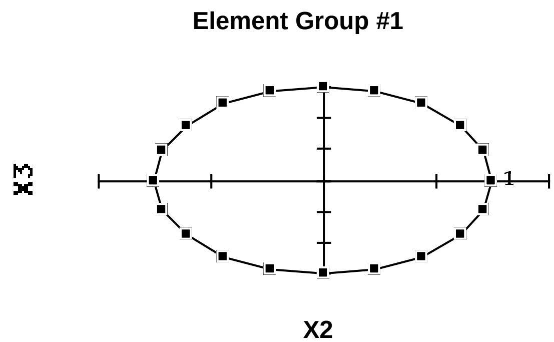


Figure 18: Cross-section plot with center points of bricks

Now select the file **NODE.XLS** and input numbers as shown in Figure 19.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R
1	node\	x	y	z	node\	1	2	3	4	5	6	node\	1	2	3	4	5	6
2	1	0.000	0.000	0.000	1	1	1	1	1	1	1	1	0	0	0	0	0	0
3	2	0.050	0.000	0.000	2	0	0	0	0	0	0	2	0	0	0	0	0	0
4	3	0.100	0.000	0.000	3	0	0	0	0	0	0	3	0	0	0	0	0	0
5	4	0.150	0.000	0.000	4	0	0	0	0	0	0	4	0	0	0	0	0	0
6	5	0.200	0.000	0.000	5	0	0	0	0	0	0	5	0	0	0	0	0	0
7	6	0.250	0.000	0.000	6	0	0	0	0	0	0	6	0	0	0	0	0	0
8	7	0.300	0.000	0.000	7	0	0	0	0	0	0	7	0	0	0	0	0	0
9	8	0.350	0.000	0.000	8	0	0	0	0	0	0	8	0	0	0	0	0	0
10	9	0.400	0.000	0.000	9	0	0	0	0	0	0	9	0	0	0	0	0	0
11	10	0.450	0.000	0.000	10	0	0	0	0	0	0	10	0	0	0	0	0	0
12	11	0.500	0.000	0.000	11	0	0	0	0	0	0	11	0	0	-0.01	0	0	0

Figure 19: Global coordinates, boundary identifiers and global forces

Then select the file **ELEM.XLS** and input numbers as shown in Figure 20.

	A	B	C	D
1	elmt. #	group #	node 1	node 2
2	1	1	1	2
3	2	1	2	3
4	3	1	3	4
5	4	1	4	5
6	5	1	5	6
7	6	1	6	7
8	7	1	7	8
9	8	1	8	9
10	9	1	9	10
11	10	1	10	11

Figure 20: Nodal connectivities of the elements

Now go back to the control part of the file **INPUT.XLS**. Complete the input by inputting numbers and answers ("yes" or "no") as shown in Figure 21. Now press the appropriate buttons as explained in Section 4. You will then have several plots and the result file **OUTPUT.XLS**.

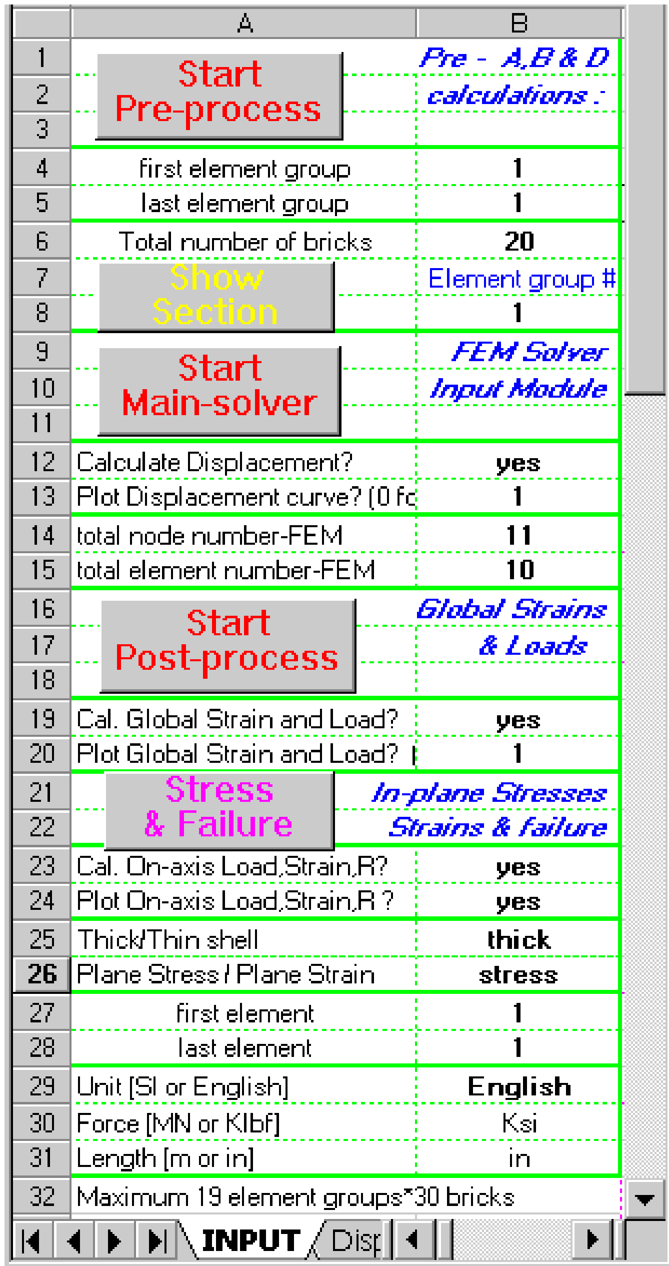


Figure 21: Control part for Case 1

5.2 Case 2

The second case is a laminated composite beam under the three point bending. Concentrated force (100 N) is applied downward at the center, whereas two equally spaced points are simply-supported. The length of the beam is 1 m and the width of the cross-section is 7 cm. Figure 22 shows this beam and its cross-section.

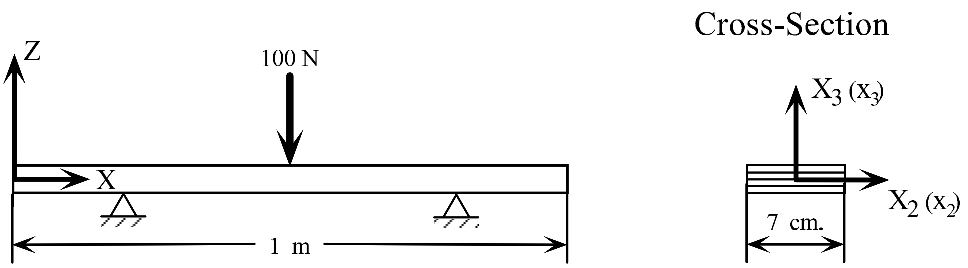


Figure 22: Laminated composite beam with subjected to three point bending

The analytic solutions for the displacement and slope are obtained using the beam theory.

$$\begin{aligned}
\text{For } 0 \le x < a\ ; \qquad & w(x) &=& \tfrac{p}{D_{11}}\,\tfrac{b^2}{16}\,(x-a) \\
& w'(x) &=& \tfrac{p}{D_{11}}\,\tfrac{b^2}{16} \\
\text{For } a \le x < a+\tfrac{b}{2}\ ; \qquad & w(x) &=& \tfrac{p}{D_{11}}\,[\,\tfrac{b^2}{16}\,(x-a) - \tfrac{1}{12}\,(x-a)^3\,] \\
& w'(x) &=& \tfrac{p}{D_{11}}\,[\,\tfrac{b^2}{16} - \tfrac{1}{4}\,(x-a)^2\,]
\end{aligned}$$

where $p = \mathrm{P}/w = -1428.57$ N/m, P $= -100$ N, $w = 7$ cm, $a = 0.2$ m, $b = 0.6$ m and D_{11} is one component of the bending stiffness matrix (i.e., [D] matrix calculated by the laminate plate theory).

Two cases are compared in this section — $[0_{10}/90_{10}]_S$ and $[60_{10}/\text{-}60_{10}]_S$ layups. Figure 23 shows the results of the layup $[0_{10}/90_{10}]_S$ with $D_{11} = 1671$ N/m. And Figure 24 shows the results of the layup $[60_{10}/\text{-}60_{10}]_S$ with $D_{11} = 246.3$ N/m.

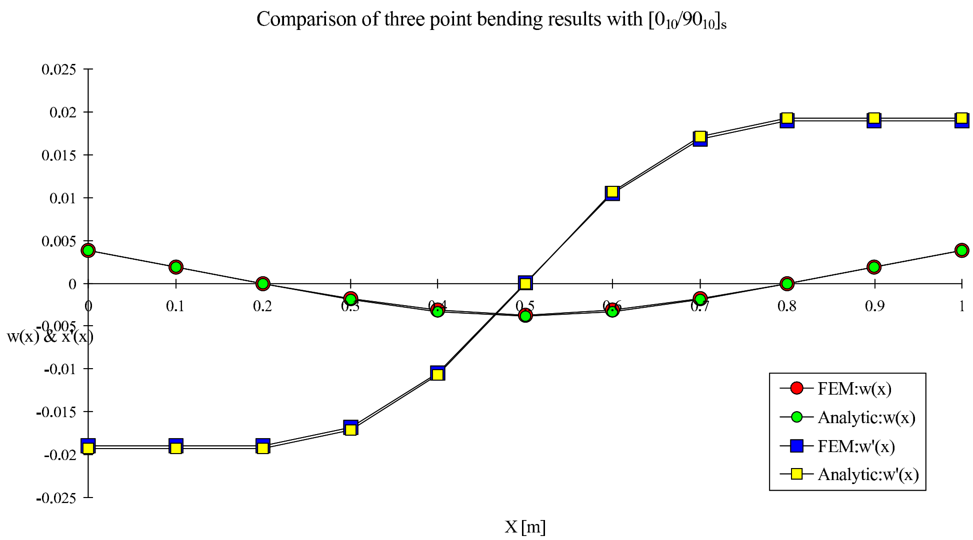


Figure 23: Displacement and slope of $[0_{10}/90_{10}]$s beam

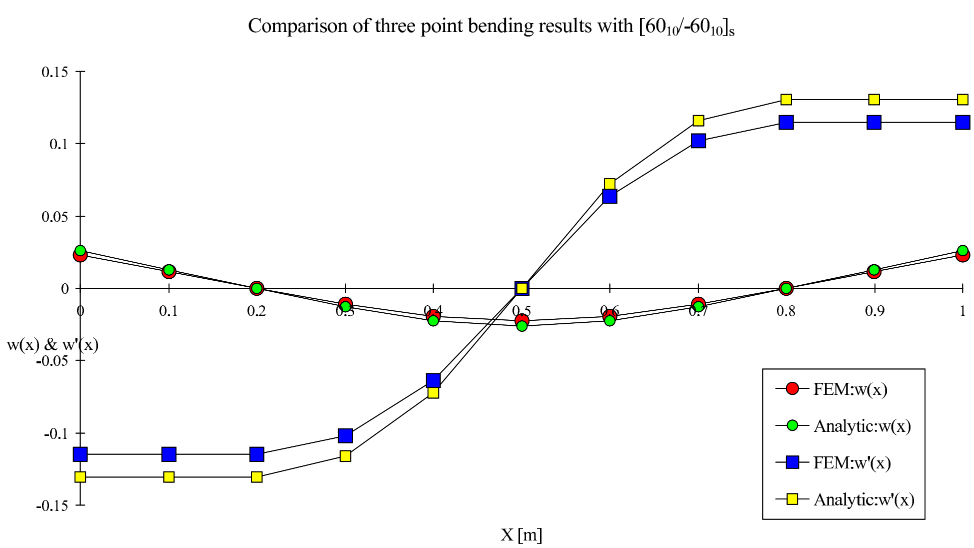


Figure 24: Displacement and slope of $[60_{10}/\text{-}60_{10}]$s beam